CHAPTER 2

Heat Transfer Enhancement by Turbulent Impinging Jets and Ribbed Channel Flows

R.S. Amano
University of Wisconsin-Milwaukee, Milwaukee, WI, USA.

Abstract

This chapter focuses a numerical and experimental study on heat transfer of turbulent air jets impinging on a flat plate. This technology has a useful application in an impinging cooling blade in a gas turbine blade and many heat exchangers. An axisymmetric air jet was positioned perpendicular to a flat plate which was uniformly electrically heated. Nozzle-to-plate spacings of 4, 7, and 10 were tested at Reynolds numbers of 10,000, 20,000, and 40,000. A symmetric air flow was ensured and confirmed by measurements. Axi-symmetric Navier–Stokes equations are solved using several different turbulence closures. The turbulent Prandtl number is proposed as a function of the local ratio of turbulent energy production to energy dissipation rate. Predictions by the present models show generally satisfactory agreement with the experimental data.

Keywords: Computational Fluid Dynamics (CFD); heat transfer enhancement; impinging jet; turbulence model

1 Introduction

1.1 Impinging jet

During the past decades, jet impingement cooling has been widely used in applications ranging from paper manufacturing to the cooling of gas turbine blades because of the very high local heat transfer coefficients that are possible. While the use of single jet impingement results in non-uniform cooling, increased and more uniform mean heat transfer coefficients may be attained by dividing the total cooling flow among an array of smaller jets. Unfortunately, when the spent fluid from the array’s central jets interact with the outer jets, the overall mean heat transfer coefficient is reduced. This problem can be alleviated by locally extracting the spent fluid before it is able to interact with the surrounding jets [1–11].
The objective of this study is to predict the heat and momentum transfer characteristics of a turbulent submerged axisymmetric jet that impinges on a flat plate. The problem is of interest to the fields of jet cutting, Vertical Take-Off and Landing (VTOL) aircraft aerodynamics, paint spraying, arc welding with a shielding gas, and a number of heating and cooling applications. The flow and heat transfer characteristics of an unconfined air jet that is impinged normally onto a heated flat plate have been experimentally investigated for high Reynolds numbers ranging from 30,000 to 70,000 and a nozzle-to-plate spacing range of 1–10 by Ozmen and Baydar [12]. The mean and turbulence velocities by using hot-wire anemometry and impingement surface pressures with pressure transducer are measured. Hammad and Milanovic [13] reported an experimental investigation that was performed to study the flow structure of a submerged water jet impinging normally on a smooth and flat surface using particle image velocimetry. The author showed a semi-confined setting provided properly characterized flow boundary conditions. The Reynolds number based on jet mean exit velocity was \( Re = 15,895 \). The pipe-to-plate separation was varied between 1 and 8 pipe diameters. The current study focused on characterizing the flow structure close to the pipe outlet, in the impingement and wall-jet regions. Statistically averaged mean and root mean square velocities are reported for a 6D wide and 1D high, near-plate, rectangular region.

Many theoretical studies have been conducted as well as experiments. Craft and Launder [14] studied a turbulent impinging behavior using four turbulence models applicable to the numerical prediction of the impinging jets discharged from a circular pipe. They comprise one k–\( \varepsilon \) eddy viscosity model and three second-moment closures. In the test cases selected, the jet discharge was two and six diameters above a plane surface orthogonal to the jet’s axis. The Reynolds numbers were \( 2.3 \times 10^4 \) and \( 7 \times 10^4 \), with the flow being fully developed at the discharge plane. The numerical predictions, obtained with an extended version of the finite-volume code, indicate that the k–\( \varepsilon \) model and one of the Reynolds-stress models (RSMs) lead to far too large levels of turbulence near the stagnation point. This excessive energy in turn induces much too high heat transfer coefficients and turbulent mixing with the ambient fluid. The other two second-moment closures, adopting new schemes for accounting for the wall’s effect on pressure fluctuations, do much better though one of them is clearly superior in accounting for the effects of the height of the jet discharge above the plate. None of the schemes is entirely successful in predicting the effects of Reynolds number.

1.2 Turbulence models

It is commonly recognized that, without modifications, conventional isotropic eddy viscosity models fail to predict the effects of noninertial forces on turbulence and stress anisotropy, whereas second-moment closures naturally account for these effects in a systematic way. In a major way, this is due to the pressure–strain correlation that plays a vital role in determining the structure of a wide variety of turbulent flows and in capturing field distributions of Reynolds stresses.
Consequently, proper modeling of this term is essential for the development of second-order closure models that have reliable predictive capabilities. The original work of Rotta [15] was the first simple model for the slow pressure–strain correlation (i.e. the part that is independent of the mean velocity gradients) that describes return to isotropy behavior of turbulence within the framework of a full Reynolds-stress closure. Over the years, further developments and novel approaches have been reported by Speziale et al. [16], and the pressure–strain correlation has been studied extensively [17–22]. The two-component limit approach of Craft and Launder [23], the elliptic relaxation RSM of Durbin (1993), and the elliptic blending RSM [24] gave very satisfactory results in predicting turbulent flow and heat transfer applications [25].

Considering that the full RSM requires solving six stress transport equations in addition to the turbulence dissipation rate equation ($\epsilon$), a simpler version of the RSM will be used in this project, that is, the algebraic stress model (ASM) [26]. In this way, the computational time and memory can be significantly reduced, while preserving all the favorable features of the RSM in the model itself. Models of this class are derived by adequate truncations of the full (differential) Reynolds-stress transport equations. The RSM to ASM truncation is based on the assumption of weak equilibrium, expressed as $\frac{Da_{ij}}{Dt} = 0$, where $a_{ij}$ is the turbulent stress anisotropy. These models provide for improved physical modeling of turbulence over traditional two-equation eddy viscosity models.

Despite many superior features, Rodi’s model is an implicit formulation often encountering undesirable numerical behavior. Gatski and Speziale [27] applied the method to compute three-dimensional flows in noninertial frames by treating the ratio of production to dissipation of turbulent kinetic energy as a known quantity and linearizing the problem. Girimaji [28] demonstrated that if one does not linearize about the equilibrium value of production to dissipation, it is possible to reduce the nonlinear problem to the solution of a single cubic equation for one of the closure coefficients. Wallin and Johansson [29] and Yoder [30] extended the model to the explicit algebraic stress model (EASM). The model was then applied to a compressible planar mixing layer and a supersonic elliptic jet. In all of their calculations, the ASM demonstrated its unique ability to predict anisotropy among the Reynolds normal stresses. For the mixing layer calculation, the ASM was found to more accurately predict the shape of the mean velocity profiles and the turbulent stresses relative to two-equation eddy viscosity models. However, both cases mentioned are unidirectional, with slow evolution of the stress anisotropy, and are thus suitable for ASM. Admittedly, ASMs have not been very successful when applied on their own to rapidly evolving, unsteady and strongly non-equilibrium flows where stress anisotropy undergoes substantial changes. In addition, to make an ASM explicit, one must introduce regularization assumptions that usually reduce the model’s credibility. The main motive for EASM is to make the model numerically more robust by avoiding possible singularities if the denominator of the ASM expression becomes too small in the course of a solution. The aforementioned shortcomings fade away if ASM is applied only to the near-wall region, as is our intention in the hybrid Large Eddy Simulation/Reynolds-Averaged Navier-Stokes (LES-RANS)
strategy. Our proposed study will focus on an ASM with an elliptic-blending model (EBM by [24, 31]) as these have not yet been tested for the ASM.

It is our view, however, that the main cause of this failure is the two-equation eddy viscosity scheme adopted in all cases to span the near-wall sublayer rather than the outer layer models on which the present study is focused.

In this chapter, an extensive numerical and experimental study of an impinging jet on a flat plate is presented with regard to heat transfer characteristics along the flat plate.

2 Experimental Study

Experimental heat transfer coefficients were obtained using an electrically heated flat plate with an axisymmetric air jet impinging normally to its surface (see Fig. 1). Nozzle-to-plate spacing, distance from the nozzle centerline, and nozzle Reynolds numbers were varied so that the heat transfer characteristics could be determined over a range of conditions.

The test section consisted of a thin stainless steel plate with the heated portion being 15 cm × 15 cm. Copper bus bars were soldered to both ends of the plate to conduct electric power to the plate. The plate was mounted on a block, and its back and sides were heavily insulated to minimize conduction losses. Five thermocouples were mounted to the back of the plate and distributed around its center 2.5 cm

Figure 1: Experimental setup.
apart along a radial line. A sixth thermocouple was located 5.0 cm from the center on a radial line 90° from the other radial line. A large (75 cm × 75 cm × 100 cm) enclosure surrounded the test section to ensure that air movements within the room did not affect the tests. Holes in the enclosure top are created so that these can allow heated air to escape, thus maintaining a relatively constant temperature within the enclosure during tests.

The nozzle consisted of a straight, circular stainless steel tube with a squared-off end (4.6 mm ID) and the tube’s length, L, from entrance to nozzle was L/D = 75. The long nozzle length ensured a fully developed velocity profile at the nozzle exit. The jet air temperature was measured with a thermocouple in the air line approximately 50 cm from the nozzle exit. The nozzle was mounted on a traversing platform that moved directly above the line of five thermocouples. By traversing along this line, a more continuous set of data for larger distances from the centerline of the jet could be obtained than if a stationary jet was used.

Before the electrical power was applied to the test section, the nozzle was positioned perpendicularly above the plate at a specified H/D at the intersection of the two radial lines on which the thermocouples were mounted, where H denotes the distance from the nozzle exit to the plate. Circumferential symmetry of the flow was checked with a velocity impact probe located 3 mm above the plate and at 2.5 and 5.0 cm from the nozzle centerline along the two radial lines. Generally symmetric flow patterns were measured, but there was a slight instability in the flow that caused a small random shifting of the flow. After the power was turned on and steady state temperatures were achieved, comparison of the measured temperatures also confirmed the symmetric flow pattern. For each test run, test section power was set such that the temperature difference between the wall thermocouple at the jet stagnation point and the air flow was about 15 K. This power was maintained throughout complete test run as the nozzle was traversed along the radial line. Power level was changed for each combination of Reynolds number and H/D.

To obtain the experimental heat transfer coefficients, a conduction heat transfer analysis for each test run (fixed Re and H/D) was performed using a finite difference code. Uniform heat generation rate, circumferential symmetry, and constant thermophysical properties were assumed; radiation from the top of the test section was estimated and found to be negligible. An adiabatic boundary condition was assumed for the sides and back. For the top surface boundary condition, a convective heat transfer coefficient distribution was assumed. A quarter cylinder was meshed in the r–θ–z directions with a total of 50,010 cells (56,672 nodes). Grid independence was verified by refining the grid systematically in the radial and axial directions by halving the size of the grids. The largest difference was less than 0.6% between the results from the 50,000-cell grid used and a grid four times as large.

The assumed heat transfer coefficient distribution was used with the power generation rate to calculate the surface heat flux distribution, surface temperatures, and backside wall temperatures, which were then compared with the experimentally measured wall temperatures. (Taking advantage of circumferential symmetry, a regression equation was obtained of the experimentally measured backside wall
temperature as a function of radial distance from the jet stagnation point.) If the measured and calculated backside wall temperatures did not agree, then a new heat transfer coefficient distribution was obtained from the calculated surface heat flux, wall and air temperature, and the procedure was repeated. This iterative procedure was used until convergence (less than 0.3°C) was obtained between the calculated and measured temperatures.

Comparing the Nusselt number distribution assuming a uniform surface heat flux versus the heat flux distribution accounting for radial conduction demonstrated that radial conduction is important within about three diameters (for Re = 40,000 and H/D = 10) to about seven diameters (for Re = 10,000 and H/D = 4) from the jet stagnation point. For distances farther than these from the jet stagnation point, there was no difference between the uniform and nonuniform heat flux solutions. At the jet stagnation point, the Nusselt numbers calculated with the nonuniform heat flux was greater than those calculated assuming a uniform heat flux by about 49% at Re = 40,000 and H/D = 10 to about 100% at Re = 10,000 and H/D = 4.

Data were taken at H/D = 4, 7, 10, Reynolds numbers of 10,000, 20,000, and 40,000, and 0 ≤ r/D ≤ 22. Heat fluxes of 1,950–8,200 W/m² were used, which resulted in temperature differences between the wall and air of about 15–75 K. Uncertainties in the experimental data are estimated in Table 1.

3 Mathematical and Physical Model

3.1 Governing equations

The theoretical part of the present work is based on the numerical solution of the two-dimensional form of the time-averaged Navier–Stokes equations, by incorporating the Boussinesq turbulent viscosity concept. Turbulent viscosity is defined by the high Reynolds number version of the k ~ ε model of turbulence [32]. The governing equation following this approach for the present flow configuration can be written in the following general form:

\[
\frac{1}{r} \left[ \frac{\partial}{\partial r} \left( r \rho U \phi \right) + \frac{\partial}{\partial r} \left( r \rho V \phi \right) \right] = \frac{1}{r} \left[ \frac{\partial}{\partial x} \left( r \frac{\partial}{x} \phi \right) + \frac{\partial}{\partial r} \left( r \frac{\rho \phi}{x} \right) \right] + S_{\phi} \tag{1}
\]

where φ stands for different dependent variables (U, V, k, ε, T) for which the equations are to be solved. All of the governing equations used in the present paper are summarized in Table 2, and the constants used in the turbulence model are given.
Table 2: Summary of equations solved.

<table>
<thead>
<tr>
<th>Equations</th>
<th>φ</th>
<th>Γ&lt;sub&gt;eff&lt;/sub&gt;</th>
<th>Sϕ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>x-Momentum</td>
<td>U</td>
<td>μ + μ&lt;sub&gt;T&lt;/sub&gt;</td>
<td>( \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[ (\mu \mu + \mu \mu_T) \frac{\partial U}{\partial x} \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[ r (\mu \mu + \mu \mu_T) \frac{\partial V}{\partial x} \right] )</td>
</tr>
<tr>
<td>r-Momentum</td>
<td>V</td>
<td>μ + μ&lt;sub&gt;T&lt;/sub&gt;</td>
<td>( \frac{\partial p}{\partial r} + \frac{\partial}{\partial x} \left[ (\mu \mu + \mu \mu_T) \frac{\partial U}{\partial r} \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[ r (\mu \mu + \mu \mu_T) \frac{\partial V}{\partial r} \right] - 2 (\mu + \mu \mu_T) \frac{V}{r^2} )</td>
</tr>
<tr>
<td>Energy</td>
<td>T</td>
<td>( \frac{\mu}{\sigma} + \frac{\mu \mu_T}{\sigma_T} )</td>
<td>( \rho G + \rho \varepsilon )</td>
</tr>
<tr>
<td>Turbulence kinetic energy</td>
<td>K</td>
<td>μ + μ&lt;sub&gt;T&lt;/sub&gt;</td>
<td>( \frac{\rho \varepsilon}{\sigma_k} )</td>
</tr>
<tr>
<td>Turbulence energy dissipation</td>
<td>ε</td>
<td>μ + μ&lt;sub&gt;T&lt;/sub&gt;</td>
<td>( C_1 \frac{\rho \varepsilon}{k} \left( G - C_2 \frac{\rho \varepsilon^2}{k} \right) )</td>
</tr>
<tr>
<td>Turbulent viscosity</td>
<td>μ&lt;sub&gt;T&lt;/sub&gt; = C&lt;sub&gt;µ&lt;/sub&gt; ρk&lt;sup&gt;2&lt;/sup&gt;/ε</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Turbulence energy generation</td>
<td>G</td>
<td>( \mu_T \left[ \left( \frac{\partial U}{\partial r} \right)^2 + 2 \left( \frac{\partial U}{\partial x} \right)^2 + 2 \left( \frac{\partial V}{\partial x} \right)^2 + 2 \left( \frac{V}{r} \right)^2 \right] )</td>
<td></td>
</tr>
</tbody>
</table>
EMERGING TOPICS IN HEAT TRANSFER

With the high Reynolds number formulation, levels of near-wall kinetic energies and dissipation rates are obtained as described in the following subsection.

3.2 Near-wall turbulence models for high Reynolds

While viscous effects on the energy-containing turbulence motions are negligible throughout most of the flow, the no-slip condition at a solid interface always ensures that, in the immediate vicinity of a wall, viscous effects will be influential. Although the thickness of this viscous-affected zone is usually two or more orders of magnitude smaller than the overall width of the flow, its effects extended over the whole flow field because, typically, 50% of the velocity change from the wall to the free stream occurs in this region.

3.3 The k–ε turbulence model

The near-wall two-layer and three-layer models are summarized by Amano [33] where a technique adopted by Chieng-Lauder for two-layer and the extension to three-layer models by Amano are compared. The main concept is, as can be seen in Fig. 2, a parabolic variation of the turbulent kinetic energy is assumed, which corresponds to a linear increase of fluctuating velocity with distance from the wall within the viscous sublayer. The turbulent kinetic energy, k, varies linearly toward the other node points. The turbulent shear stress is zero within the viscous sublayer, and the shear stress undergoes an abrupt increase at the edge of the sublayer while varying linearly over the remainder of the cell. The details of this treatment of k equation are described elsewhere [34]. In the present study, the model
introduced by Amano [35] was further applied to the present impinging jet study. Here the treatment of the ε equation in the near-wall cell is developed considering that ε near the wall is an order of magnitude larger than that in the fully turbulent core and reaches its maximum at the wall. Each term in the ε equation should be evaluated in accordance with the k equation rather than being approximated under local equilibrium conditions.

In the viscous sublayer and in the fully turbulent region, turbulence energy, k, and energy dissipation rate, ε, are expressed with the notation in Fig. 2. The details of this two-layer model are given in Amano [34] and the summary of the k–ε two-layer near-wall model is listed in Table 4.

The coefficients, a and b, are given as follows:

\[ a = k_p - \frac{k_p - k_N}{y_p - y_N} y_p \]
\[ b = \frac{k_n - k_v}{y_n - y_v} \]

3.4 The Reynolds stress turbulence model

When the RSM is used, the form of the governing momentum equation is different from eqn (1). Because of unmanageable number of terms, the tensor notation is used. The momentum equation is different from eqn (1). Owing to the unmanageable number of terms, the tensor notation is used. The momentum equation is as follows:

\[ \frac{\partial}{\partial x_j} \left( \rho U_j U_j \right) = - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho \frac{\partial U_i}{\partial x_j} \right] \]  

(2)

The set of differential equations governing the transport of the kinematic Reynolds stresses, \( u_i u_j \), can be written as

\[ \frac{\partial}{\partial x_k} \left( \rho U_k u_i u_j \right) = \left( \rho u_j u_k + \rho u_i u_k \frac{\partial U_i}{\partial x_k} \right) \frac{\partial U_i}{\partial x_k} \frac{\partial U_j}{\partial x_k} \]  

(3)

To close eqn (3) in terms of mean velocities and the Reynolds stresses, the turbulence quantities on the right-hand side must be represented by empirical functions of mean velocities, the Reynolds stresses, and their derivatives. In this study, attention was focused on the approximation of the third group of terms, namely the pressure–strain correlation. The pressure–strain correlation has a turbulence energy redistributive effect; thus, this correlation mainly controls the energy exchange among the Reynolds-stress components.
Table 4: Different models of wall boundary.

<table>
<thead>
<tr>
<th>Generation rate in k-equation G</th>
<th>Models used</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1) [ \frac{\tau_w (U_n - U_v)}{y_n} + \frac{\tau_w (\tau_n - \tau_w)}{\rho K^* k_{v}\sqrt{1/2} y_n} \left( 1 - \frac{y_v}{y_n} \right) ]</td>
<td>KE1</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>+ \left[ \tau_w \left( 1 - \frac{y_v}{y_n} \right) + \frac{\tau_n - \tau_w}{2} \left{ 1 - \left( \frac{y_v}{y_n} \right)^2 \right} \right]</td>
</tr>
<tr>
<td>(2) [ \frac{\tau_w (U_n - U_v)}{y_n} ]</td>
<td>KE3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Dissipation rate in k-equation ( \varepsilon )</th>
<th>Models used</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1) [ \frac{2k_{v}^{3/2}}{y_n R_v} + \frac{1}{y_n C_{\ell}} \left[ \frac{2}{3} \left( k_{n}^{3/2} - k_{v}^{3/2} \right) \right] ]</td>
<td>KE1</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>+ 2 a \left( k_{n}^{1/2} - k_{v}^{1/2} \right) +</td>
</tr>
<tr>
<td>(2) [ C_{\mu}^{3/4} k_{p}^{3/2} U f \left( \sqrt{\tau_w / \rho y_p} \right) ]</td>
<td>KE3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Generation rate in k-equation ( C_{2} G \dot{e}^{2} / k )</th>
<th>Models used</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1) [ \frac{\tau_w}{\rho} \frac{C_{1}}{\frac{1}{2} k_{v}^{1/2} C_{\ell} Y_{v}} \left[ \tau_w \left( \frac{k_{n}^{1/2} - k_{v}^{1/2}}{y_n} + \frac{b \lambda}{2</td>
<td>a</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>+ \left[ \tau_n \tau_w \left( \frac{k_{n}^{1/2} - k_{v}^{1/2}}{y_n} \right) +</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>+ \frac{C_{1}}{C_{\ell} y_n} \tau_w \left( 2 \left( k_{n}^{1/2} - k_{v}^{1/2} \right) +</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>+ \frac{2}{3} \frac{\tau_n - \tau_w}{y_n} \frac{1}{b} \left( k_{n}^{3/2} - k_{v}^{3/2} \right) \frac{\partial V}{\partial x}</td>
</tr>
<tr>
<td>(2) Not evaluated</td>
<td>KE3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Dissipation rate in k-equation ( \varepsilon )</th>
<th>Models used</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1) [ C_{2} \left[ \frac{12 \tau^{2} k_{v}}{y_{v} y_n} + \frac{1 - y_{v}/y_n}{C_{\ell}^{2}} \left( \frac{a^{2}}{y_{v} y_n} + \frac{2 a b \ln y_n}{y_{v}^{2}} + b^{2} \right) \right] ]</td>
<td>KE1</td>
</tr>
<tr>
<td>(2) Not evaluated</td>
<td>KE2</td>
</tr>
</tbody>
</table>

KE1 KE2 KE3
The authors have tested several models of the pressure–strain correlation and discovered that it was necessary to incorporate double-stress products into these correlations for the computations of highly turbulent flows. This idea is certainly consistent with the form possessed by the third-order moments that appear in the diffusion of eqn (3); that is, the fourth-order tensors (quadruple terms) are expanded into products of the Reynolds stresses by employing the Gaussian assumption.

Generally, the pressure–strain correlation is divided into two parts: one with fine-scale turbulence effect only, and the other with both fine- and large-scale turbulence eddies. The first part with the fine-scale turbulence effect was formulated by Rotta [36] in the form of anisotropy. The second term has been proposed by Naot et al. [37] as follows:

\[ p \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) = C_{\phi 2} \left( G_{ij} - \frac{2}{3} \delta_{ij} G \right) \]  

(4)

where

\[ G = \rho u_i u_j \frac{\partial U_i}{\partial x_j} \]  

(5)

and

\[ G_{ij} = \left( \rho u_i u_k \frac{\partial U_j}{\partial x_k} + \rho u_j u_k \frac{\partial U_i}{\partial x_k} \right) \]  

(6)

The formulation of the pressure–strain correlation can be performed by formulating a Poisson equation for the fluctuating pressure, \( p \), after taking the divergence of the equation for the turbulent fluctuating velocity, \( u_i \), which can be re-expressed to write the pressure–strain correlation as follows:

\[ p \frac{\partial u_i}{\partial x_j} = S_{ij} + \frac{1}{4\pi} \int_{\text{vol}} \left( T_{ij}^1 + T_{ij}^2 \right) \frac{d\text{Vol}}{R} = S_{ij} + \phi_{ij}^1 + \phi_{ij}^2 \]  

(7)

where

\[ T_{ij}^1 = \rho \left( \frac{\partial^2 u_i / u_m}{\partial x_\ell / \partial x_m} \right) \frac{\partial u_i}{\partial x_j} \]  

(8)

\[ T_{ij}^2 = 2\rho \left( \frac{\partial U_\ell / u_m}{\partial x_m} \right) \frac{\partial u_m}{\partial x_\ell} \frac{\partial u_m}{\partial x_\ell} \]

\[ R = |x - y| \]

where \( S_{ij} \) denotes a surface integral and is important in the near-wall region, whereas \( \phi_{ij}^1 \) and \( \phi_{ij}^2 \), respectively, represent the Rotta term and the redistribution term, which is responsible for mean gradients. In this study the model developed by Amano et al. [38] was employed.
The pressure–strain correlation without the near-wall effects are given by

\[
\psi^1 + \psi^2 = -C_{\phi_1} \rho \left( \frac{\varepsilon}{k} \left( \frac{u_i u_j - 2}{3} \delta_{ij} \right) + (7 C_{\phi_2} - 10) \right) \frac{u_i u_j - 2/3 S_{ij}}{k} \]

\[
-2 (C_{\phi_2} - 1) \left[ \left( G_{ij} - \frac{2}{3} S_{ij} G \right) + \left( H_{ij} - \frac{2}{3} \delta_{ij} G \right) \right] - \frac{2}{5} \rho (2 C_{\phi_2} - 1) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) k
\]

where

\[
H_{ij} = \left( \frac{\rho u_i u_k \partial U_k}{\partial x_j} + \frac{\rho u_j u_k \partial U_k}{\partial x_i} \right)
\]

3.5 Heat transfer model

Different models used in the present computation are summarized in Table 4. In this table, the second cases of generation rate and destruction rate in \( \varepsilon \) equation are not given, because in models 2 and 3 the dissipation rate \( \varepsilon \) is given a fixed value of \( \varepsilon = k \rho^{3/2} / C_p T \) in the near-wall cells.

The interlinkage used between the near-wall variation in temperature and the local wall heat flux is given as follows:

\[
\frac{\rho c_p (T - T_w) k_p^{1/2}}{q'_w} = \frac{U_p k_p^{1/2}}{\tau_w / \rho} + P
\]

where \( P \) is the \( P \) function given as [39]

\[
P = 9.24 \left[ (\sigma / \sigma_T)^{0.75} - 1 \right] + 0.28 \exp (-0.007 \sigma / \sigma_T)
\]

In this equation, \( \sigma_T \) is the turbulent Prandtl number. Two different expressions for \( \sigma_T \) are used in this paper: one is a constant value of 0.9, the other is a function of the local ratio of turbulent energy generation to dissipation [40].

\[
\sigma_T = 0.225 (5.4 + G / \varepsilon) / (1.2 + G / \varepsilon)
\]

4 Numerical Solution Procedure

The method adopted for solving eqn (1) is the solution procedure of the finite volume method in which the value of each scalar quantity is associated with every grid node (i.e. the points where the grid lines intersect), although the vector quantities (velocity components) are displaced in space relative to the scalar
quantities $p$, $T$, $k$, $\varepsilon$, and $u_iu_j$. Such a staggered grid system is advantageous in solving the velocity field as the pressure gradients are easy to evaluate and velocities are most conveniently located for the calculation of convective fluxes. It also suppresses an oscillating solution of flow fields. The QUICK scheme was used for the computations of the momentum equations due to the robustness of the equations. A hybrid scheme of convection–diffusion string is adopted in which a central-difference approximation is used when the intercell Peclet number is less than 2 and an upwind difference of the combined effects of convection and diffusion is used at the stronger level of convection (Peclet number is greater than 2) for the turbulence equations. However, there is no guarantee that the resultant velocity field will satisfy the continuity equation. Two problems of determining the pressure distribution and satisfying continuity were resolved using SIMPLEC [41].

At the outflow boundary region the flow is considered to be of the parabolic type; at the inflow region (at the nozzle exit), a uniform velocity profile is given and the turbulence intensity ($k/\bar{U}_N^2$) is assumed to be in the range of 0.005 based on consideration of the experimental apparatus used and along the centerline of the jet, symmetry is used.

5 Results and Discussion

5.1 Impinging jets

In the present experimental data a second maximum in the heat transfer coefficient curve was not detected by Narayanan et al. [42] who observed it for the cases of short nozzle-to-plate distance ($H/D < 2$), which lies in either transitional and/or potential-core jet impingement. The second maximum may depend strongly on the level of turbulence promoter to change the turbulence intensity level at the nozzle exit; it is possible that the turbulence present was not sufficient to cause the second maximum to occur.

Figure 3 shows the computed contour of the turbulence intensity. The turbulence level is higher in the recirculating region due to a highly shearing action of

![Figure 3: Turbulent kinetic energy contours.](figure3)

(a) $H/d=4$   (b) $H/d=10$
the impinging jet, which transmit a high turbulence toward the stagnation point. Figure 4 demonstrates the pressure contour in the impinging region. The highest pressure is observed at the stagnation point.

Figures 5–7 compare the measured and calculated distributions of heat transfer coefficient along the flat plate for nozzle-to-plate distance, \( H/D = 4 \) and 10. For both cases, the nozzle Reynolds number is \( 2 \times 10^4 \). The calculations were made for the three different wall boundary treatments given in Table 4. Note the
Figure 6: Heat transfer coefficient distribution along the plate.

Figure 7: Heat transfer coefficient distribution along the plate.
generally close similarity of the results for the two different values of $H/D$; in particular, the calculated results show a much steeper slope in the region where $r/D$ is less than 3. KE1 evaluates the mean generation and dissipation rates in both the $k$ equation and $\varepsilon$ equation. KE2 evaluates the mean generation and dissipation rate only in the $k$ equation and fixes the $\varepsilon$ value at the wall with the functional relation $\varepsilon = k^{3/2}/C_P \tau$. KE3, the simplest case, does not take the $k$ value nor the turbulent shear stress into account in the evaluation of mean generation and dissipation rates in either the $k$ or $\varepsilon$ equation. The RSM in the figures represent the model with the full Reynolds-stress equations. Comparing KE1, KE2, KE3, and RSM, it is obvious from Figs 5–7 that RSM improves the prediction of the maximum heat transfer coefficient by 30%. However, the slope of the heat transfer coefficient curve near the maximum heat transfer point stays the same. Comparing KE1, KE2 and RSM, a large difference between these two models appears at the stagnation point. KE1 predicts the maximum heat transfer coefficient 10–12% closer than KE2 for both $H/D = 4$ and 10. This indicates that the small changes in the $\varepsilon$ equation can significantly cause different results especially in the low Reynolds number region where very fine-scale eddy motions take place. Consequently, such fine-scale motions directly influence the behavior of $\varepsilon$ which is a fine-scale property.

Although refinement of the model used to predict $k$ and $\varepsilon$ in the near-wall region can improve the prediction of the heat transfer characteristics, there is still a 10–25% difference between the measured and computed results at the stagnation point. There are some reasons that may cause these discrepancies. One of them may come from the $\varepsilon$ equation per se. It would be unreasonable to have the turbulence energy generation rate $P$ appear in any true dissipation equation because, at high Reynolds numbers, the small-scale dissipative motions should have no direct link to the large-scale motion. It is equally implausible that the Boussinesq effective viscosity should depend on a fine-scale property such as $\varepsilon$ because momentum transport in turbulence is predominantly related to large-scale motions. From this argument it can be suggested that both the $k$ and $\varepsilon$ equations be reevaluated so that the $k$ and $\varepsilon$ equations are solved for two different time scales to model medium- and large-scale motions. However, if we note the asymptotic values, the agreement between the measured and computed values is excellent. This indicates that for most of the high Reynolds number flow, the standard $k$ and $\varepsilon$ model is still superior to the other turbulence models not only for its reliability and accuracy but also for its simplicity.

As expected, RSM gives better results in comparison with the computations using $k$–$\varepsilon$ models with any near-wall models. This is the evidence that although RSM requires more CPU time, the nonisotropic effect of turbulence can be evaluated properly with the second-order closures.

Another possible explanation for the disagreement between the measured and computed results may come from the interlinkage between momentum and heat transfer. Knowledge of the Prandtl number is always used as a means of finding the turbulent effective diffusivity from an already computed value of turbulent effective viscosity rather than vice versa. There is thus, inevitably, the tendency to look to changes in the heat-transfer mechanism to explain variations in the turbulent
Prandtl number that may actually occur from one point in the flow to another. The effect of the turbulent Prandtl number on the heat transfer is shown in Fig. 8 for nozzle Reynolds number of $2 \times 10^4$ and $H/D = 7$. In this figure, the computed heat transfer for different turbulent Prandtl numbers are compared with the experimental results. In one case, the constant Prandtl number of 0.9 solving with KE1, represented by TPR1, is used and, in the other, the functional expression of the local ratio of turbulent energy generation to dissipation, which was proposed by Morris et al. [40] [eqn (14)] solved by KE1, represented by TPR2, is used. These results are also compared with the RSM computations with eqn (14). It is apparent from this figure that the difference between these two cases appears only at the stagnation point. This result corresponds approximately with Chua et al. [43] in which the measured turbulent Prandtl number is 0.6 on the axis of the jet, falls to about 0.4 in the region of maximum generation, then rises to near unity toward the edge of the jet. It appears, then, that the second-order closure for modeling the scalar equations, which can take the variation of local generation to dissipation ratio into account, should give better prediction near the centerline of the jet.

Another possible reason for the disagreement near the stagnation point is radial conduction. The reported experimental heat transfer coefficients were calculated from the measured surface temperatures and an average heat flux over the plate. Radial conduction would cause heat flow toward the stagnation point. Thus, the experimental heat transfer coefficient would increase at the stagnation point and
would decrease farther away from the stagnation point if the actual local heat flux could be determined. However, the required boundary conditions needed to solve the elliptic differential equation are not all known. The effect of the radial conduction plays a role as long as the thickness of the plate is not negligibly thin compared with its diameter. There is also the experimental uncertainty that can cause the discrepancies between the measured and calculated heat transfer levels.

Finally, the dependence of the heat transfer coefficients on the jet Reynolds number is shown in Fig. 9. In this figure, the maximum heat transfer coefficients are plotted as a function of nozzle Reynolds number for different nozzle-to-plate distances. Although the levels of heat transfer coefficients of both measured and computed seem to agree better for H/D = 4 than for H/D = 7 and 10, the dependence of Reynolds number is worse for H/D = 4. The experimental data are proportional to \( \text{Re}_D^{0.8} \) for every case, as are the numerical results for H/d = 7 and 10. However, the calculated data for H/D = 4 display a \( \text{Re}_D^{0.7} \) dependence. This shows that the computed results for smaller nozzle-to-plate distances (H/D = 4) become less dependent on Reynolds number than the experimental results. For the case of a short nozzle-to-plate distance (H/D = 4), most of the high Reynolds number turbulence models may not be able to predict heat transfer coefficients accurately near the stagnation points because the length of the potential core of the jet is longer than the nozzle-to-plate distance.

### 5.2 Ribbed-channel flows

Combinations of the impinging jet and channel flow cases are demonstrated as shown in Fig. 10, where the geometry of the channel used in the simulation for both cases of 90° and 45° ribs angles is shown. The dimensions of the model used for numerical simulations and experiment are shown in Table 1. The inlet of the geometry has the same inlet Reynolds number as the experiment, \( \text{Re} = 10,000 \). One side of the geometry has both ribs and bleeding holes. The other side of the geometry has only ribs.

The contours of normalized Nusselt number distribution of the holed wall, for the case of the 90° and 45° ribbed channels, are shown in Figs 11 and 12, respectively, for the \( \text{k–}\omega \), \( \text{k–}\varepsilon \), and RSM models, respectively. The resulting Nusselt number by simulation was normalized with respect to the empirical formulation of the Nusselt number for nonribbed channel determined by the Dittus-Boelter correlation

\[
\text{Nu}_0 = 0.023\text{Re}^{0.2}\text{Pr}^{0.4} \quad (20)
\]

The experimental Sherwood number was normalized with respect to the Sherwood number for the nonribbed channel. The contour distribution shows that high values appear close to the bleeding holes and low values distribute slightly away from the holes. The obtained values of the holed wall shear stress by the simulation were used to calculate the heat transfer coefficient and determine the Nusselt number distribution. The computations by \( \text{k–}\omega \), \( \text{k–}\varepsilon \), and RSM are compared with the experimental observation by Ekkad et al. [44]. It seems, as expected,
Figure 9: Maximum heat transfer coefficient as a function of Reynolds number.
that both k–ω and RSM show better match with the experimental results. These observations are also validated quantitatively for the averaged Nusselt number distributions along two bent channels as shown in Figs 13 and 14.

5.3 Large eddy simulation

Because actual heat transfer in real applications is affected by heat conduction inside the pin-fin and convective heat transfer from the pin-surface, heat transfer performance of pin-fin channels should be evaluated under conjugate heat transfer conditions. So far, most of numerical studies on conjugate heat transfer between turbulent air flow and solid metal have been solved with RANS models, because the solid has much larger heat capacity than the fluid in most applications and, therefore, the temperature of solid part is determined by the time-mean local convective heat transfer rate on the solid surface. However, a RANS-based
Figure 11: Nusselt number contour on 90° ribbed channel.

Figure 12: Nusselt number contour on 45° ribbed channel.
method can be valid only when time-mean velocity and temperature fields are obtained with a sufficient accuracy by RANS models. Thus, it is difficult to apply conventional RANS-based methods to conjugate heat transfer problems in complex turbulent flow and heat transfer fields, which are seen in pin-fin channels. Recent development of computer hardware has made computations based on the large eddy simulation (LES) has come feasible for many industrial heat transfer applications. Figure 15 shows sample computations of flow over ribs in a blade channel cooling passage. As can be observed, the computations of LES are 3D and unsteady in its nature. Therefore, it requires considerable computational CPU and the requirement of mesh size is approximately 10–100 times more than RANS approach.
The main conclusions to emerge from the present study on the turbulent impinging jet may be summarized as follows:

1. The near-wall turbulence model that was developed to evaluate the $\varepsilon$-equation generally improves the prediction near the maximum heat transfer coefficient.
2. Incorporation of the second-order closure in scalar equations can improve the prediction near the stagnation point.
3. The maximum heat transfer coefficient increases with Reynolds number. This pattern has been shown to agree with computed results for the nozzle-to-plate distance higher than 7 nozzle diameters.

Figure 15: Large eddy simulation of flows over ribs through a channel.

6 Conclusion

The main conclusions to emerge from the present study on the turbulent impinging jet may be summarized as follows:

1. The near-wall turbulence model that was developed to evaluate the $\varepsilon$-equation generally improves the prediction near the maximum heat transfer coefficient.
2. Incorporation of the second-order closure in scalar equations can improve the prediction near the stagnation point.
3. The maximum heat transfer coefficient increases with Reynolds number. This pattern has been shown to agree with computed results for the nozzle-to-plate distance higher than 7 nozzle diameters.
In conclusion, one of the important facts is that while the Boussinesq approach with two-equation turbulence models gives relatively good agreement with experiments, significant improvements in predictions with this model can be obtained by incorporating the second-order closure model for the flow computations of the impinging jet. This improvement should be performed before extending the turbulence model to a multiequation model in which all the stress components must be computed in their own transport equations. Furthermore, the development of the scalar flux equations can easily improve the flow prediction.

Although refinement of the model used to predict \( k \) and the \( \varepsilon \) prediction of the heat transfer characteristics in the near-wall region can be improved, there is still a 10–25% difference between the measured and computed results at the stagnation point. There are some reasons that may cause these discrepancies. One of them may come from the \( \varepsilon \) equation per se. It would be unreasonable to have the turbulence energy generation rate appear in any true dissipation equation because, at high Reynolds numbers, the small-scale dissipative motions should have no direct link to the large-scale motion. It is equally implausible that the Boussinesq effective viscosity should depend on a fine-scale property such as \( \varepsilon \) because momentum transport in turbulence is predominantly related to large-scale motions. The RSM, which presented the largest improvement in the prediction of the heat transfer rates, is the one of the promising approach for turbulent heat transfer. This is mainly because the nonisotropic effect occurring in the impingement and accelerating flow regions can be approximately be taken into account through a reasonable mechanism of the redistributive energy treatment operated by the pressure–strain correlation.

In 21st century, as the computer hardware is advanced much, it has become feasible to extend the simulation technique toward LES and DES, which can significantly improve the predictions of the turbulent flows.

References


